International Journal of Mechanical and Production
Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 6, Issue 2, Apr 2016, 23-36

CFD ASSISTED NOZZLE DESIGN IN BURR REMOVAL PROCESS

VEERALKUMAR THAKUR[1], TARUN YADAV[2] & RAJIV B[3]

[1,3]Department of Production and Industrial Engineering, College of Engineering, Pune, Maharashtra, India

[2]AURDC, Hindustan Aeronautics Limited, Nasik Aircraft Division, Nasik, Maharashtra, India

ABSTRACT

The paper deals with comparative study of different types of nozzles and selection of best suitable nozzle was done; which can be used for metal chip blowing purpose during machining in CNC machine. Three different type of nozzles, one Convergent type, one Bent Convergent& one Convergent-divergent (C-D) type, were designed and modeled in CATIA V5 software. These nozzles were designed at par with the various commercially available blow off nozzles. Further, Computational Fluid Dynamics (CFD) analysis of these nozzles were carried out for estimating the various internal flow properties like variation of density, temperature, pressure, velocity, mass flow rate and Mach number. The chocking phenomenon in nozzles is also studied in this paper. After analysis, it has been concluded that the modeled Convergent Nozzle is best for blowing metal chips during machining in CNC machine. However, the modeled Bent Convergent and C-D nozzles would not be effective for the blow off purpose, this could be due to the choking and reverse flow at the exit. Further, this convergent nozzle is manufactured, validated and found satisfactory.

KEYWORDS: Nozzle, Computational Fluid Dynamics, Chip Blowing, ANSYS Fluent

NOMENCLATURE

Mass flow rate	$\dot{m}$
Pressure	P
Temperature	T
Area	A
Mach number	M
Velocity	V
Ratio of specific heats	γ
Density	ρ
Subscripts used	
Total or static	T
Exit or output	E
Throat	t

Superscript '*' is used to denote chocking conditions

Received: Mar 18, 2016**; Accepted:** Mar 28, 2016**; Published:** Apr 02, 2016**; Paper Id.:** IJMPERDAPR20163

Original Article

INTRODUCTION

Advances in aviation industries is one of the most important aspect, from the country's defense point of view. Aviation industries are mainstay of a country's Air Force. Improvement in productivity of an aviation industry benefits the country's defense. Improvement in productivity for a manufacturing industries stands for decrease in time required for machining of parts during production, sub assembly and assembly of components. Further, automation of any manufacturing process would lead to enormous increase in production rate. In our case, it was done by automation of the metal chips cleaning process in 5-axis CNC aluminum profiler machine. This was achieved by modelling and analysis of 3 different nozzles through CFD.

CFD is a branch of fluid dynamics that deals with computerized numerical methods and algorithms to simulate and evaluate problems that are related to different fluid flow. Simulation of the interaction of liquids and gases with surfaces constrained by boundary conditions are carried out with computers. CFD instead of being used chiefly as a design authentication software, can be used in the design process to drive geometry change. CFD can provide the accuracy that cannot be achieved by custom modelling means based on assumptions and simplified numerical techniques and empirical formulae. Inherent advantages of CFD are obtained for e.g. offers fast and inexpensive solutions compared to experimental solutions and more accurate then empirical methods used for designing. Precise simulation of flows through the nozzles is vital for estimation of velocity, temperature, Mach number and pressure values.

The CNC machine considered in this paper, was the 5-axis twin spindle aluminum profiler CNC machine. It is installed in one of manufacturing shop. This machine has the capacity of performing high speed machining in 5 different axis and is generally used for profiling of aluminum plates. The CNC machine produces large amount of continues metal chips/burrs during machining of Aluminum parts due to its ductile property. Further, these metal chips were resulted in accumulation of heavy amount of metal chips on the machined part; which made machining difficult. That is why, cleaning of these metal chips/burrs is required every hour that meansCNC machine to be on standby after every hour. General practice is to clear it manually by using human labor. This process had a major impact on production rate, as it increases machining time as the machine is ideal during cleaning of metal chips and in turn decreases productivity.

The main purpose of this study was to design and develop a 'nozzle' which would amplify the air velocity to blow off the metal chips quickly.

The available input parameters are as follows:

Inlet Pressure for Nozzle = 6 Bar (measured through pressure gauze)

Inlet temperature for Nozzle = 297 K (measured through temperature sensor)

LITERATURE REVIEW

The CFD researchers and practitioners have carried out multiple research in field of nozzle design and analysis in various fields. *Gamble et al.* [1]provides guidelines and procedures for incorporating requirement for features such as thrust vectoring and reversing considerations into the design of gas turbine exhaust nozzles. *Mohan Kumar G et al.*[4] focuses on designing a de Lavel nozzle to attain supersonic flow and optimizing it to achieve maximum thrust without flow separation due to Shock waves. *Prafulla et al.*[8] conducted CFD analysis of flow within, Convergent–Divergent rectangular supersonic nozzle and supersonic impulse turbine with partial admission have been performed. Analysis had been

performed according to shape of a supersonic nozzle and length of axial clearance, with an objective of investigating the effect of nozzle-rotor interaction on turbine's performance. *G. Satayanarayana et al.*[2] performedCFD analysis of flow within Convergent-Divergent supersonic nozzle of different cross sections rectangular, square and circular. The analysis had been performed according to the shape of the supersonic nozzle and keeping the same input conditions. Their objective was to investigate the best suitednozzle which gives high exit velocity among the different cross sections considered.

NOZZLE DESIGN AND MODELLING

The word 'Nozzle' derives from the word 'nose', meaning 'small spout'. A Nozzle is a device with varying cross-sectional area of its profile and with an objective of converting the low velocity to high velocity, high pressure to low pressure and affecting other parameters. Fundamentally, it has the ability for converting pressure energy into kinetic energy and the thermal energy also transforms into kinetic energy and results in temperature drop at the output and this related to the linear momentum producing thrust. Parameter density decreases and Mach number increases. One of the most critical tasks regarding nozzle is its analysis using computational fluid dynamics (CFD); so that various internal flow parameters like velocity gradient, pressure gradient, temperature change, velocity distribution through various sections of the nozzle, changes in Mach number of the fluid etc. can be analyzed

There are different types of nozzle designs possible. Types of the nozzles discussed in this paper are:

- Convergent type.
- Bent-converging type.
- Convergent-divergent (C-D) type.

Geometry Conversion

By the method of characteristics which were assumed for the output parameters, the following convergent and C-D nozzles geometries were generated. The bent converging nozzle was referred from *G. Satyanarayan et al.* [2]and was designed as required.

Convergent Type of Nozzle

This type of nozzle has profile of decreasing area. Mach number is M<1. As in this type density does not fluctuate much at low input pressures of the fluid. This type of nozzle profile is generally used to accelerate the output fluid. Figure 1 shows the 3-D surface model of convergent nozzle in Catia V5 with its flow direction. In figure 2, the 2-D geometric profile of the convergent nozzle is shown with its profile co-ordinates. The co-ordinates of end-points of the nozzle are given in Table 1. This type of nozzle exhibits the isentropic relations of thermodynamics, referred from *Isidoro Martinez, "Nozzles", 1995-2015*[7]in which mass flow rate is:

$$\dot{m} = \gamma\rho A = \sqrt{\frac{\gamma}{R}}\frac{p}{\sqrt{T}}\,\mathrm{MA}$$

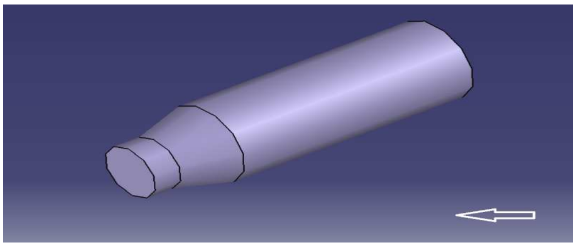

Figure 1: Convergent Nozzle with Flow Direction

Table 1: Co-Ordinates of Convergent Nozzle

Geometry Points	X (mm)	Y (mm)
1	0	0
2	17.5	0
3	22.5	1.5
4	25	1.5
5	25	7.5
6	22.5	7.5
7	17.5	9
8	0	9

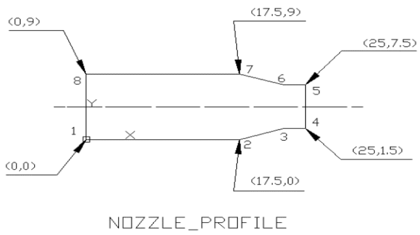

Figure 2: Profile of Convergent Nozzle

Bent-Converging Type of Nozzle

The profile was generated from the curved profile which would give the direction and amplify the fluid and an area to the flow of fluid at the output and increase the velocity of the fluid due its decreasing area. The bent contour was generated with designing co-ordinates shown in figure 4 by giving the constraints of length, height and output area of the nozzle. A surface model was developed in Catia V5 shown in figure 3. The output is a rectangular area and referred from *G. Satyanarayan et al.*[2].

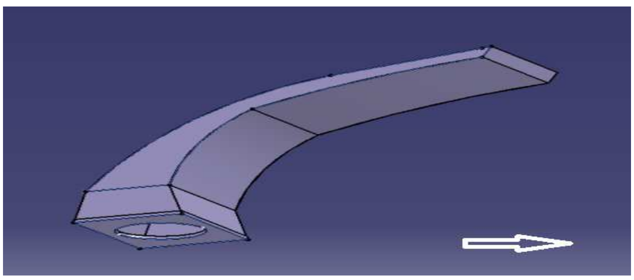

Figure 3: Bent-Convergent Nozzle with Flow Direction

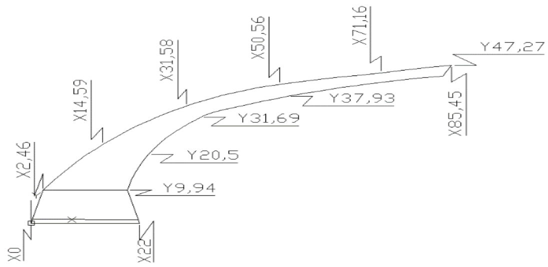

Figure 4: Profile of Bent-Convergent Nozzle

Convergent-Divergent Type of Nozzle (C-D)

Converging-diverging nozzle is also known as *De Lavel nozzle or 'Condi' nozzle (Developed by Swedish inventor Gustaf de Lavel)*, as the name states, it has a smooth converging- diverging profile with a smooth small straight section call as throat between converging and diverging section. This is the only nozzle that gets supersonic at M>1. Usually it first converges and then diverges. But could be designed as per requirement. The fluid may accelerate to its maximum speed at M<1 or M=1 (i.e. choked) at the throat. Variation of Mach number in this nozzle is shown in figure 5. Then based on the 2-D profile designed in figure 7, a surface model of the C-D nozzle was generated in Catia V5 as shown in figure 6. This type of profile is needed for impact on a large area and output thrust of the fluid. For the purpose of chip blowing, this profile of the nozzle is also suitable. At the output Mach number increases than one i.e. M>1, nozzle goes into supersonic zone.

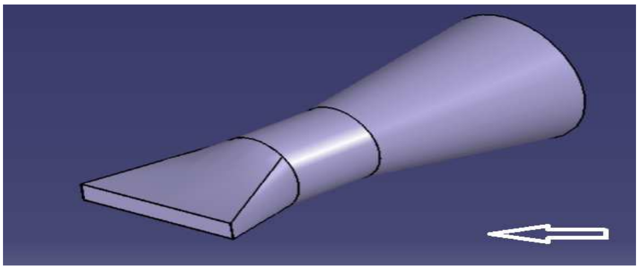

Figure 5: Convergent-Divergent Nozzle with Flow Direction

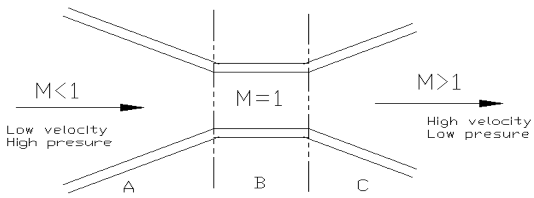

Figure 6: Variation of Mach number through C-D nozzle

Chocking Phenomenon in Nozzles

An effect of compressible flow in which the sonic velocity has been reached i.e. the subsonic velocity reaches M=1 and in which the mass flow rate is maximum, is often said choking conditions in the nozzle. The flow rate increases with a decrease in the pressure ratio (P^*/P) and attains the maximum value of the critical pressure ratio (P^*/P) = 0.54 for air. When the (P^*/P) is further reduced the nozzle will be in chocking conditions. The mass flow rate through the nozzle has reached its maximum possible value, the choked value. Thereafter mass flow rate remains constant.

$$\frac{P^*}{P} = \left(\frac{1+\frac{(\gamma-1)}{2}M_E{}^2}{\frac{(\gamma+1)}{2}}\right)^{\left(\frac{\gamma}{\gamma-1}\right)}$$

Put $\frac{T^*}{T_t} = \left(\frac{P}{P_T}\right)^{\left(\frac{\gamma}{\gamma-1}\right)} \frac{P^*}{P_T} = \left(\frac{2}{\gamma+1}\right)^{\left(\frac{\gamma}{\gamma-1}\right)}$

All the above equations are isentropic equations and referred from *Isidoro Martinez, "Nozzles", 1995-2015*[7].So, basically it can be understood that, when Mach number increases above a specified value, there is generation of a reverse shock wave at the tip of the nozzle. And this phenomenon is responsible for the loss of energy and decrease in mass flow rate.

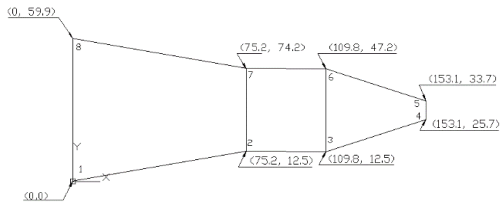

Figure 7: Profile of C-D Nozzle

Mathematical Analysis of Parameters

- **Bent Converging Type of Nozzle**

Length = 80mm

Inlet area = 452.16 mm^2

Outlet area = 30x3 mm^2(rectangular)

Inlet temperature = 291.7 k

Outlet temperature = 259.93 k

Inlet Pressure = 666715.88 pascal

Outlet pressure = 146610.82 pascal

γ =1.4

Neglecting bent losses we can treat it as a convergent type nozzle and approximate required values.

Let P_1 = pressure of fluid at the point 1,

V_1 = velocity of fluid at point 1 (≈ 0),

T_1 = temperature of fluid at point 1,

ρ_1 = density of fluid at point 1,

AndP_2,V_2, T_2 and ρ_2 are corresponding values of pressure, velocity, temperature and density at point 2.

Then velocity of fluid at throat, point 2, for nozzle can be expressed as,

$$V_2 = \sqrt{\frac{2\gamma}{(\gamma-1)}\frac{P_1}{\rho_1}\left[1-\left(\frac{P_2}{P_1}\right)^{\frac{\gamma-1}{\gamma}}\right]}$$

Above equation is referred from *Isidoro Martinez, "Nozzles", 1995-2015*[7].

$P_1 = \rho_1 RT_1$

$666715.88 = \rho_1 * (432) * (291.7)$

$\rho_1 = 5.290\ \mathrm{kg/m^3}$

$$V_2 = \sqrt{\frac{2*1.4}{(1.4-1)}\frac{666715.88}{5.290}\left[1-\left(\frac{146610.82}{666715.88}\right)^{\frac{1.4-1}{1.4}}\right]}$$

$= \sqrt{7 * 126033.24 * [0.35126]}$

= 556.68 m/s

Similarly, the other nozzle calculations were done as shown below.

- **Converging- Diverging Type of Nozzle**

Length	= 30mm
Inlet area	=452.16 mm^2
Outlet area	= 30 x 3 mm^2 (rectangular)
Inlet temperature	=292.57 k
Outlet temperature	=288.4489 k
Inlet Pressure	=672950.69 pascal
Outlet pressure	= 174899.88 pascal

$P_1 = \rho_1 RT_1$

$672950.69 = \rho_1 * (432) * (292.5)$

$\rho_1 = 5.365\ \mathrm{kg/m^3}$

$$V_2 = \sqrt{\frac{2*1.4}{(1.4-1)}\frac{672950.69}{5.365}\left[1-\left(\frac{174899.88}{672950.69}\right)^{\frac{1.4-1}{1.4}}\right]}$$

$= \sqrt{7 * 125433.49 * [0.3195]}$

= 529.65 m/s

- **Converging Type of Nozzle**

Length	= 41 mm
Inlet area	= 254.34mm^2
Outlet area	= 113.04mm^2

Inlet temperature =296.87 k

Outlet temperature =237.34 k

Inlet Pressure =708265.25 pascal

Outlet Pressure = 384317.90 pascal

From equation (1),

$$V_2 = \sqrt{\frac{2\gamma}{(\gamma-1)}\frac{P_1}{\rho_1}\left[1-\left(\frac{P_2}{P_1}\right)^{\frac{\gamma-1}{\gamma}}\right]}$$

$P_1 = \rho_1 RT_1$

$708265.25 = \rho_1 * (432) * (296.87)$

$\rho_1 = 5.522\ \mathrm{kg/m^3}$

$$V_2 = \sqrt{\frac{2*1.4}{(1.4-1)}\frac{708265.25}{5.522}\left[1-\left(\frac{384317.90}{708265.25}\right)^{\frac{1.4-1}{1.4}}\right]}$$

$= \sqrt{7 * 128262.45 * [0.1599]}$

= 378.89 m/s

Computational Fluid Dynamic Analysis

Geometry

Table 2: Grid Details

Sr. No.	Nozzle Geometry	Grid Type	Number of Elements (Millions)	Number of Nodes (Millions)
1.	Bent-convergent	Fine	4.2	0.9
2.	C-D	Fine	2.5	0.6
3.	Convergent	Fine	1.5	0.4

The three types of nozzle geometries i.e. Bent-converging nozzle, Convergent-Divergent nozzle and Convergent nozzle were modelled in CATIA V5. The principle difference among these three nozzles were their different configurations and applications. Further, these geometries were imported in ICEM CFD 12.0 software for meshing.

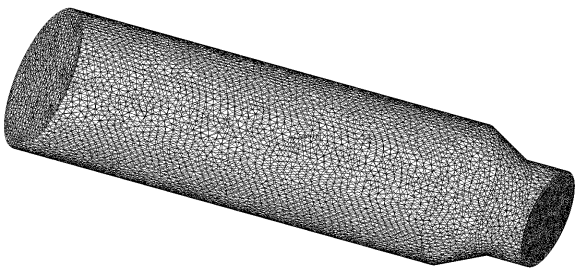

Figure 8: Surface Mesh of Computational Domain Convergent Nozzle

Mesh Generation

The fine surface and volume mesh for all three type of nozzles were generated in ICEM CFD 12.0 software. All meshes are hybrid mesh made with the combination of triangular elements, tetra elements, and prism layers. Table.2 shows the mesh details of various grids. Figure 8, 9, and 10 shows the wire frame of surface mesh of, Convergent nozzle, Bent-converging and C-D nozzle respectively. The grids were sufficiently fine near the walls, away from the wall and boundaries to capture the exact flow physics. The prism layers were generated near the walls so that the viscous behavior and turbulence of internal fluid can be accurately predicted. Figure 11 represents the cut section of a volume mesh, where prism layer are generated at the wall of a bent-convergent nozzle. Similar type of prism layers are also generated in Convergent and C-D type nozzle. After meshing, CFD simulations in ANSYS FLUENT 12.0 were carried out on all grids to analyze the flow behavior of various nozzles.

BOUNDARY CONDITIONS

The internal flow analysis was carried out by running simulations in commercial software ANSYS Fluent 12.0 version. The dimensional units in Fluent were set to SI.

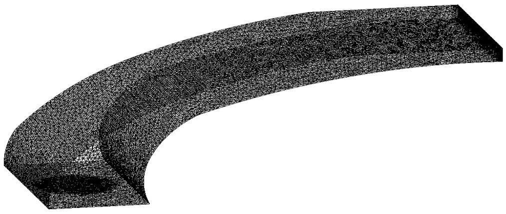

Figure 9: Surface Mesh of Computational Domain and Bent-Convergent Nozzle

Material Properties

The test case is a compressible flow problem where the value of Mach number varies from inlet to exit. Ideal gas was considered as the material. The various gas properties taken are as follows:

- Density = Ideal gas density
- Specific Heat capacity (Cp) = 1006.43 J/Kg.K
- Viscosity = 1.7894 e-5 Kg/m-s
- Thermal conductivity = 0.0242 W/m.K
- Molecular weight = 28.966 kg/kgmol.

Following are the boundary conditions which were set before running the simulation:

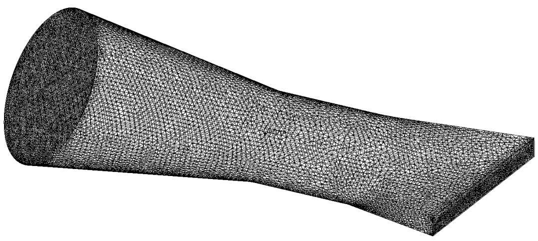

Figure 10: Surface Mesh of Computational Domain and C-D Nozzle

- **Nozzle Inlet**

Nozzle inlet was set as a pressure inlet, where gauge total pressure was taken as 6 bar. Inlet temperature was set to 297 K.

- **Outlet of the Nozzle**

Outlet pressure was set to atmospheric pressure i.e. 1 bar and outlet temperature was set to 0 K.

- **Wall**

The nozzle was considered as a stationary wall with no slip condition

- **Control Setup and Discretization**

The Density based solver with k-e turbulence model and energy equation on was considered for simulations. In solution controls, the Turbulence Dissipation rate, Turbulence Kinetic Energy, and Turbulence Viscosity were set to 0.8, 0.8 and 1 respectively. The under relaxation factors were considered as default value. In solution methods, the implicit formulation with flux type Roe-FDS was set. The least cell square gradient with second order upwind discretization, turbulent kinetic energy and turbulence dissipation rate was considered.

- **Initialization**

The solution was initialized with hybrid initialization. The convergence criteria was set to 10-3.

COMPUTATIONAL FLUID DYNAMIC RESULTS

Residual Plot

Figure 12 shows the residual plot of convergent nozzle. It is noticed that residuals are dropped up to 10^{-3} in 2000 iterations and after that solution becomes stable. This signifies that solution is converged. The Bent nozzle and C-D nozzle residuals are dropped up to 10^{-3} in 1800 iterations and after that solution become stable.

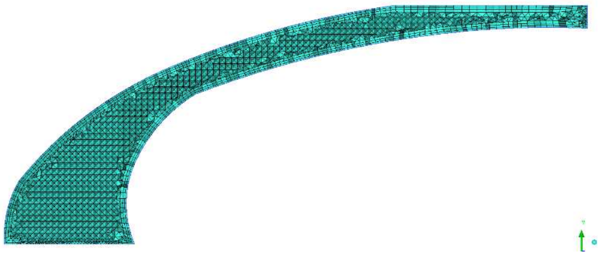

Figure 11: Prism Layer at Boundries

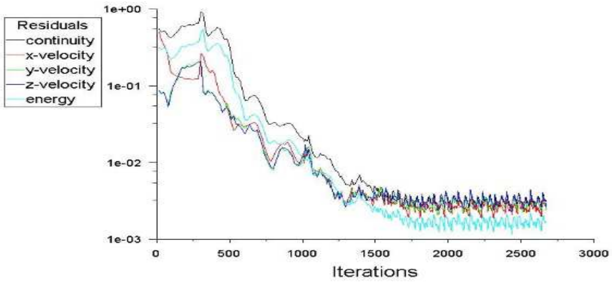

Figure 12: Residual plot of Convergent Nozzle

PRESSURE CONTOUR PLOT

Figure 13, 14 and 15 shows the contour pressure plot of Convergent nozzle, Bent Convergent and Convergent-Divergent nozzle respectively. The pressure found decreased gradually from inlet to outlet throughout the length of the nozzle. This confirms the physics requirement, as stated by Bernoulli's that drop in pressure lead to increase in velocity along the expansion zone. An uneven drop in pressure up to 50000 Pascal at exit is noticed in both Bent-converging and Convergent-Divergent type nozzles. This has arisen due to formation of heavy shock waves, which in turn lead to choking and reverse flow at exit. Hence efficiency of these nozzles got decreased. The pressure drop in convergent type nozzle was found satisfactory as per isentropic relation.

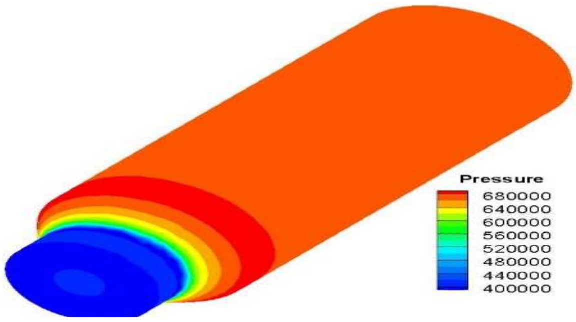

Figure 13: Pressure Contour of Convergent Nozzle

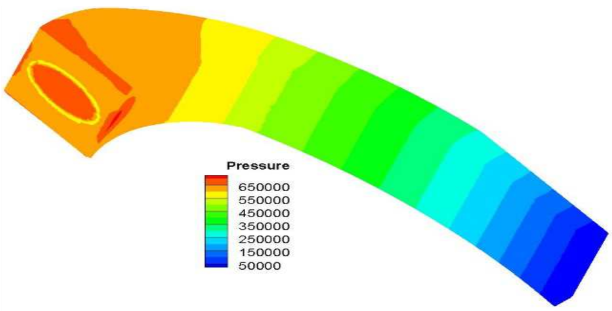

Figure 14: Pressure Contour of Bent-Convergent Nozzle

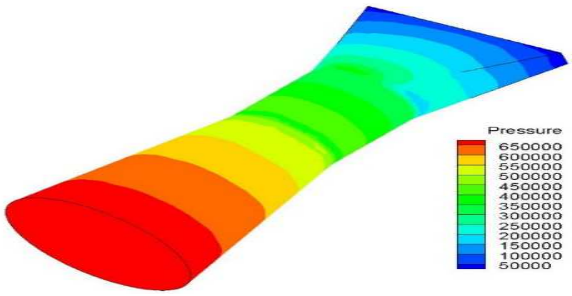

Figure 15: Pressure Contour of C-D Nozzle

Mach Number Contour Plot

The Mach number contour plot of various nozzle are shown in figure 16, 17 and 18. It is visualized that Mach number is gradually increasing from inlet to exit. The flow at exit is found supersonic (M>1) in Bent and Convergent-Divergent type nozzles while in Convergent nozzle it was found subsonic (<1).

The Mach number near the wall is high due to turbulence and viscosity of the fluid. In C-D type nozzle, the Mach number in convergent area and throat is found same.

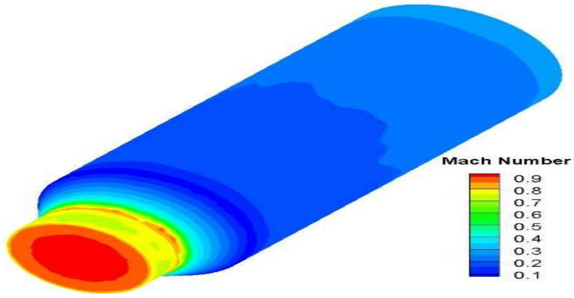

Figure 16: Mach Number Contour of Convergent Nozzle

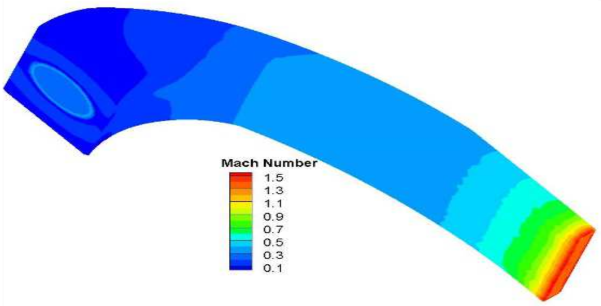

Figure 17: Mach Number Contour of Bent-Convergent Nozzle

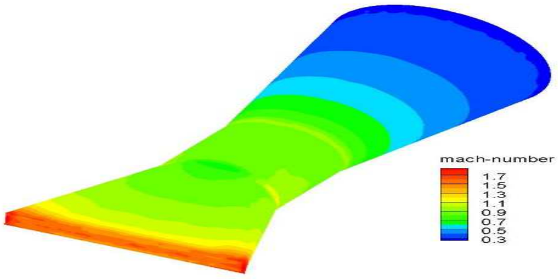

Figure 18: Mach Number Contour of C-D Nozzle

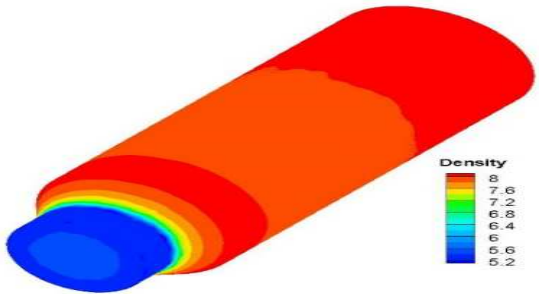

Figure 19: Density Contour of Convergent Nozzle

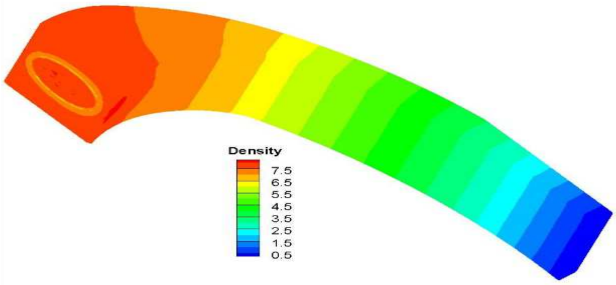

Figure 20: Density Contour of Bent-Convergent Nozzle

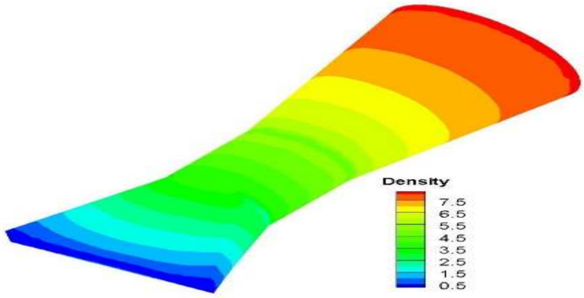

Figure 21: Density Contour of C-D Nozzle

Density Contour Plot

The density contour plot of various nozzle are shown in figure 19, 20 and 21. The density dropped gradually from inlet to exit throughout the nozzle length. It is visualized that density varies from 7.5 to 0.5 in C-D nozzle and bent nozzle and 8 to 5.2 in convergent nozzle C-D nozzle Mach number is gradually increasing from inlet to exit. This proves the Bernoulli's theory as per equation where density and pressure are inversely proportional to velocity of fluid in compressible flows.

Validation

The CFD results of Fluent for various nozzles were validated by comparing the values of exit velocity (V2) obtained from CFD with the value obtained from mathematical formulae. The exit velocity values of nozzles were found almost same with the CFD exit velocity values. Further, the convergent type nozzle was manufactured and results were found in good agreement with the CFD results.

ACKNOWLEDGEMENTS

Authors are greatly thankful to Hindustan Aeronautics Limited, Nasik division and Production Engineering Department of College of Engineering, Pune-411005 which were of support during their research.

RESULTS AND CONCLUSIONS

The aim of this paper was to develop an efficient air amplification device i.e. a nozzle, fit for aluminum chip blowing purposes in 5 axis CNC machine with the help of CFD analysis. Following are the conclusions:

- The mass flow rate at inlet and exit were found different for Bent-Convergent and Convergent-Divergent nozzle. Hence, it can be concluded that Chocking has occurred due to varying cross-section and it would cause reverse flow at exit which in turn would drop velocity drastically. However, the mass flow rate at inlet and exit were

found same for Convergent nozzle.

- From the results, it is found that Bent-Convergent nozzle gives exit velocity of 556.68 m/s, Converging-Diverging nozzle gives 529.65 m/s and Converging nozzle gives 378.89 m/s.
- So, compared with the Bent-Convergent nozzle and Convergent-Divergent nozzle, the velocity of Convergent nozzle was 31.93% and 28.46% less respectively. But due to chocking in Bent-Convergent and Convergent-Divergent type nozzle, we opted Convergent nozzle for blow off purpose.
- Velocity of nozzle increases at the cost of pressure drop. From analysis it can be seen that Bent-Converging nozzle gives pressure drop of 78.01%, the Convergent-Divergent nozzle gives drop of 74.01% and Convergent nozzle gives 45.73% drop. So, more pressure losses in C-D and Bent-Convergent nozzles.
- The output Mach number of Bent-Convergent nozzle was found out to be 1.423, Convergent-Divergent nozzle was 1.56 and Convergent nozzle was 0.957.
- The temperature drop in bent-convergent nozzle was 10.91%, in convergent-divergent nozzle was 1.41% and in convergent nozzle was 20.05%.
- From above results and analysis, the best nozzle suitable for chip blowing operation in CNC machine is Convergent nozzle.

REFERENCES

1. *Eric Gamble, Dwain Terrell and Rich DeFrancesco, "Nozzle Selection and Design Criteria", theAmerican Institute of Aeronautics and Astronautics, 40th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit 11-14 July 2004.*
2. *G. Satyanarayan, Ch. Varun and S.S. Naidu, "CFD Analysis of Convergent-Divergent Nozzle", ACTA Technica Corviniensis-Bulletin of Engineering Tome VI (year 2013)- Fascicule 3[July-September] 2013, ISSN 2067-3809.*
3. *Pourmahmoud, N., et. al "Computational Fluid Dynamics Analysis of Helical Nozzles Effects", Thermal Science, Year 2012, Vol. 16, pp. 151-166.*
4. *Mohan Kumar G, Dominic Xavier Fernando, R. Muthu Kumar, "Design and Optomization of De Lavael Nozzle to Prevent Shock Induced Flow Separation", Advances in Aerospace science and Applications, Volume 3, Number 2 (2013), pp. 119-124.*
5. *R. K Rajput, "Engineering Thermodynamics".*
6. *Pardhasaradhi Natta, V.Ranjith Kumar, Dr. Y.V.Hanumantha Rao, "Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd)", International Journal of Engineering Research and Applications(IJERA), Vol. 2, Issue 5, September-October 2012,pp. 1226-1235.*
7. *Isidoro Martinez, "Nozzles", 1995-2015.*
8. *B.Krishna Prafulla, Dr. V. Chitti Babu, P.Govinda Rao, "Cfd Analysis of Convergent-Divergent Supersonic nozzle", International Journal of computational Enggineering Research, Vol. 3, Issue 5, May-2013.*
9. *Gutti Rajeswara Rao, U.S. Ramakanth, A. Lakshman, "Flow Analysis in aConvergent-Divergent Nozzle using CFD", International Journal Of Research in Mechanical Enggineering, Vol. 1, Issue 2, Oct-Dec 2013, pp. 136-144.*
10. *MD. Safayet Hossain, Muhammad Ferdous Raiyan, Nahed Hassan Jony, "Comparative Study of Supersonics Nozzles", International Journal of Research in Engineering and Technology, Vol. 3, Issue: 10, Oct-2014, ISSN: 2321-7308.*